NUWC-NPT Technical Memo 15-118

Computational Fluid Dynamics of the Boundary Layer Characteristics of a Pacific Bluefin Tuna

David J. Daily
Kimberly M. Cipolla
Brian K. Amaral
Sensors and Sonar Systems Department

18 September 2015

NEWPORT

Naval Undersea Warfare Center Division
Newport, Rhode Island

ABSTRACT

The mechanism by which tuna achieve very fast swimming speeds is not presently understood, and may involve delay of transition or an advanced propulsion mechanism (or both). The issue of whether the boundary layer on a tuna swimming at typical speeds (1 to 2 body lengths/sec) is laminar, turbulent, or transitional is an open question.

Using an arc-length Reynolds number (Re_L) to estimate the nature of the boundary layer and predict when transition occurs only serves as a rough approximation. Uncertainties include the surface roughness of the skin, local favorable and adverse pressure gradients, and discontinuities such as the open mouth or juncture at the fins.

The primary objectives of this project are to compute the approximate lateral location at which transition to turbulence occurs on the tuna for various swimming speeds, and to determine the maximum speed at which laminar flow is retained on the tuna's body. Two-dimensional (2D) and three-dimensional (3D) computer models are used to compute the boundary layer characteristics and predict the lateral location of turbulence onset. The computations cover speeds ranging from 2 to 22 m/s.

ADMINISTRATIVE INFORMATION

This report was prepared under Section 219 Research Project, "Characterization of the Boundary Layers on Full-Scale Bluefin Tuna," principal investigator Dr. Kimberly M. Cipolla (Code 1512) and associate investigators Dr. David Daily and Brian Amaral. Work was performed under NWA 300000099899/0010. The sponsoring activity is the Naval Undersea Warfare Center (Code 00X).

The technical reviewer for this report was Georges Dossot (Code 1522).

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

LIST OF ILLUSTRATIONS (Cont'd)

LIST OF TABLES

LIST OF ABBREVIATIONS AND ACRONYMS

2D	Two Dimensional
3D	Three Dimensional
AUV	Autonomous Underwater Vehicle
CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
FEA	Finite Element Analysis
IGES	Initial Graphics Exchange Specification
NSWC	Naval Surface Warfare Center
NUWC	Naval Undersea Warfare Center
ONR	Office of Naval Research
PIV	Particle Image Velocimetry
RANS	Reynold-Averaged Navier-Stokes
Re_L	Arc-Length Reynolds Number
TKE	Turbulent Kinetic Energy

1. INTRODUCTION

In 2013, a research effort was initiated (funded by the Office of Naval Research (ONR)) to study the hydrodynamics of the Pacific Bluefin tuna as part of a collaboration with other research efforts around the country. The initial results are documented in reference 1. In FY-14, the project continued under internal research funding and involved measurements of drag and fluctuating wall pressure in a small tow tank at the Naval Undersea Warfare Center (NUWC) Division Newport (reference 2). The goal of this ongoing investigation is to answer three questions, two of which involve the boundary layer characteristics of the Bluefin tuna:

1. Does laminar or turbulent boundary layer separation occur on a tuna?

2. Does the transition from a laminar boundary layer to a turbulent boundary layer take place on a tuna at normal swimming speeds?

3. For the condition of a tuna gliding at an angle of attack, is there vortex shedding occurring that will impact the flow noise field?

ONR has continued funding experiments that are designed to measure boundary layer transition using particle image velocimetry (PIV) techniques along the streamwise axis of a rigid tuna model. Experiments were completed in September 2015 in Naval Surface Warfare Center (NSWC) Carderock's 36-inch water tunnel. These data provide information regarding how the tuna are able to swim so persistently and efficiently at high Reynolds numbers.

To support and supplement the experimental research, a computational fluid dynamics (CFD) model was created to determine additional details of the flow field and extend the speed range of the measurements. The CFD model was intended to estimate the location of boundary layer transition in order to reduce overall experimental research time required in the water tunnel and prevent excess data collection. This report outlines the specific steps taken to create a CFD model of a tuna and to document the results of the computational research. While the primary objectives are to compute the approximate lateral location at which a transition to turbulence occurs on the tuna for various swimming speeds and to determine the maximum speed at which laminar flow is retained on its body, some additional goals include:

1. Quantifying the mean and fluctuating velocity components

2. Quantifying the boundary layer parameters used for scaling

3. Quantifying the effects of three-dimensional flow

2. TECHNICAL APPROACH

The technical approach can be subdivided into three steps:

1. Generate 2D and 3D computational models and meshes from the computer-aided design (CAD) model.

2. Compute the flow field using an appropriate CFD solver, parameter values, and boundary conditions.

3. Analyze the data to find the pertinent information about the boundary layer physics.

2.1 GENERATING THE MODELS AND MESHES

2.1.1 CAD Model

A juvenile Pacific Bluefin tuna was harvested from the Tuna Research and Conservation Center in Monterey, California. The tuna was flash frozen and sent to a taxidermist to make a mold of the tuna, which was digitized with a laser scanning system. That scan was then exported into a format compatible with the software package Solid Works (see figure 1). Note that this computer model only contains the caudal fin (excluding the pectoral, dorsal, and anal fins and finlets). The focus of the present study was to characterize the boundary layer on the body of the fish rather than any secondary flow effects such as vortex shedding; the fins were not expected to have a significant effect on the boundary layer dynamics.

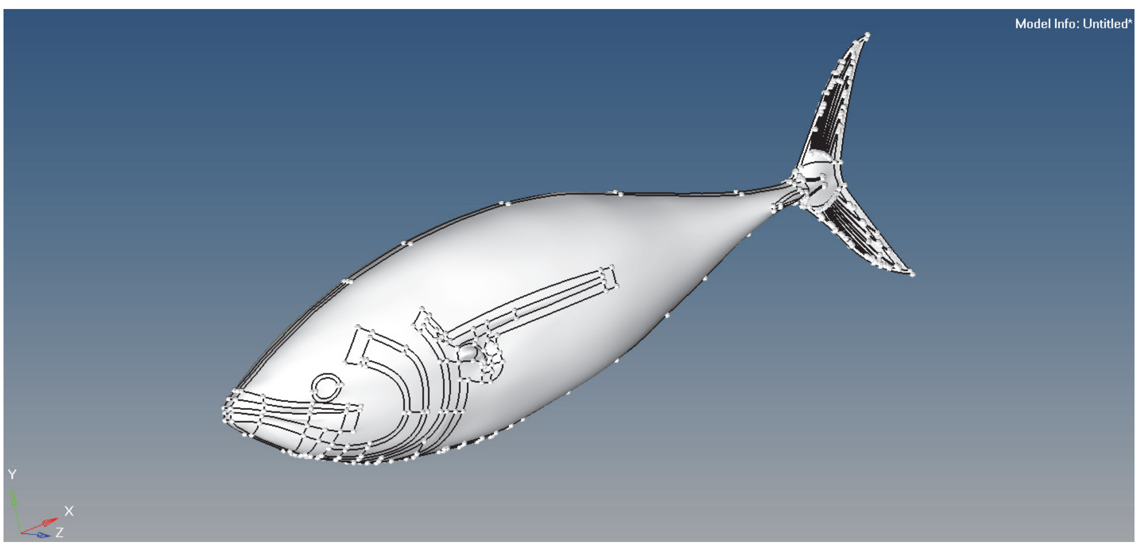

Figure 1. CAD Model of the Pacific Bluefin Tuna Harvested from the Tuna Research Conservation Center

An initial step in understanding the flow field past a tuna was to quantify the boundary layer along the centerline of the tuna model. This simplified the computation of the flow field but required the creation of a two-dimensional (2D) model of the tuna. Since creating a 2D

cross-sectional cut of the CAD model in Solid Works is a cumbersome process, the full three-dimensional (3D) model was exported to a different software package called Hypermesh to extract the cross-sectional outline. The steps for that process are outlined in section 2.1.3.

The CAD model was originally generated in English units of inches. Most CFD codes default to metric but can be changed to English units, if needed. To simplify the modeling process, the full 3D model of the tuna was exported from Solid Works in inches as an initial graphics exchange specification (IGES) file and was later imported and scaled in Hypermesh to be in metric units.

2.1.2 Surface Mesh

Meshing the surface of the CAD model was performed primarily to check the model and ensure that there were no gaps or discontinuities. Hypermesh is a meshing software package that was specifically written for finite element analysis (FEA) solvers, but in recent years it has made strides in improving its CFD meshing capabilities. While some CAD software packages can export mesh and geometry directly from a generic geometry format, such as .stl files, using this approach in Solid Works yielded a mesh that was insufficient for use in CFD applications. An image of the .stl model and mesh generated using this approach is shown in figure 2.

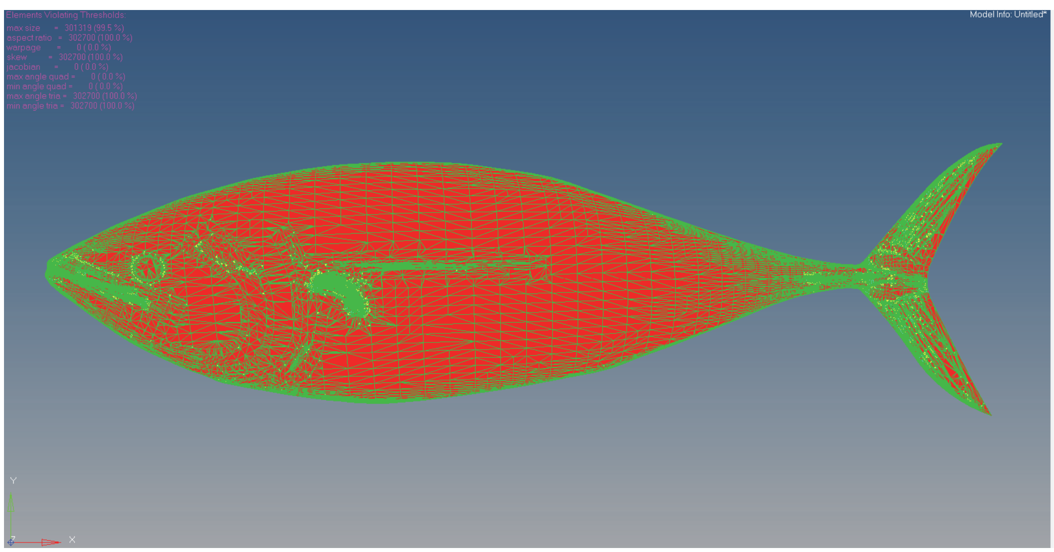

Figure 2. Mesh Created by Solid Works on the Surface of the Tuna Model. The cells highlighted in red were indendified by Hypermesh as poor quality for CFD purposes.

The cells highlighted in red were automatically flagged as being potentially problematic for their size, aspect ratio, or orientation to the direction of flow. It was determined that Solid Works was unsuitable for generating a high quality CFD mesh and therefore an alternate approach was attempted that utilized an .iges file as the default file export scheme. Most meshing software packages, like Hypermesh, prefer generic geometry file formats (e.g. .iges, .stl files) as opposed to typical CAD format files (.nx, .sldpt). When the .iges file was read and the Hypermesh algorithms were applied to the model's surface, the resulting mesh was superior to

that of the Solid Works mesh (see figure 3). Specifically, all of the cells are uniform in size and shape with no cells flagged by the software as potentially problematic.

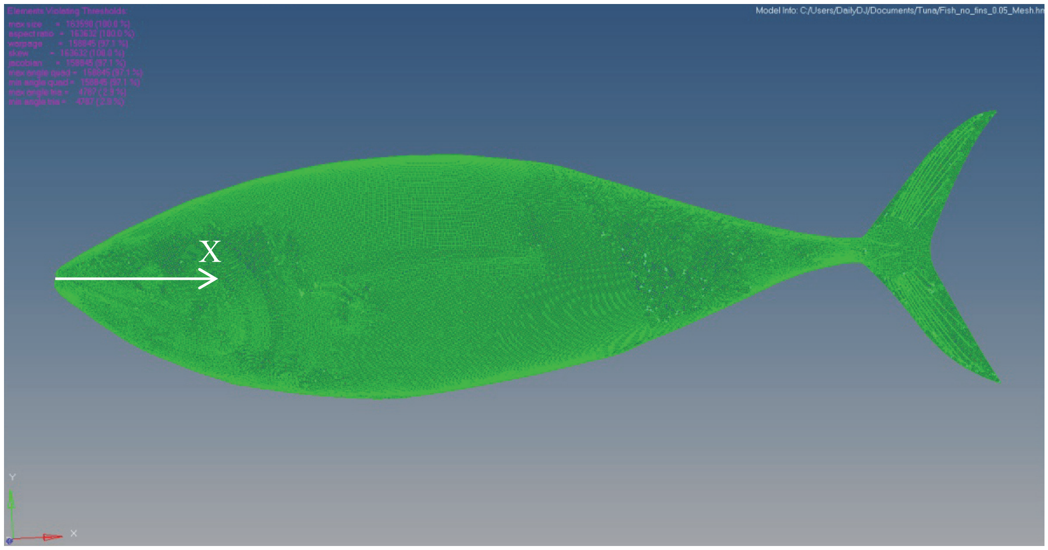

Figure 3. Mesh Generated on the Tuna Model in Hypermesh

2.1.3 2D Mesh

A 2D cross section of the tuna model was created by making a plane that bisected the tuna model along the medial line of the fish. From the CAD model, the center of mass of the tuna model was set to (0,0,0), thus a plane was created in the XY dimension that intersected this origin. This plane was used to cut the model in half and, with all other surfaces of the tuna model removed, a 2D cross section of the model remained. The steps for doing this in Hypermesh are outlined below (these steps assume the user has a basic understanding of Hypermesh).

f2
Delete all solids

Geometry>Surfaces
y-axis B = 0,0,0 Size = 6

Geometry>Surface Edit
Trim with surfs/plane
with surfs
1-select tuna surfaces
2-select bisceting plane
check "trim both" radio button

Tool>Rotate
Surfs> select surface
x-axis B = 0,0,0
angle = 90

The final step outlined above indicates that the plane of interest was rotated so the the x- and y-axes were changed to follow the convention used in Fluent (see figure 4).

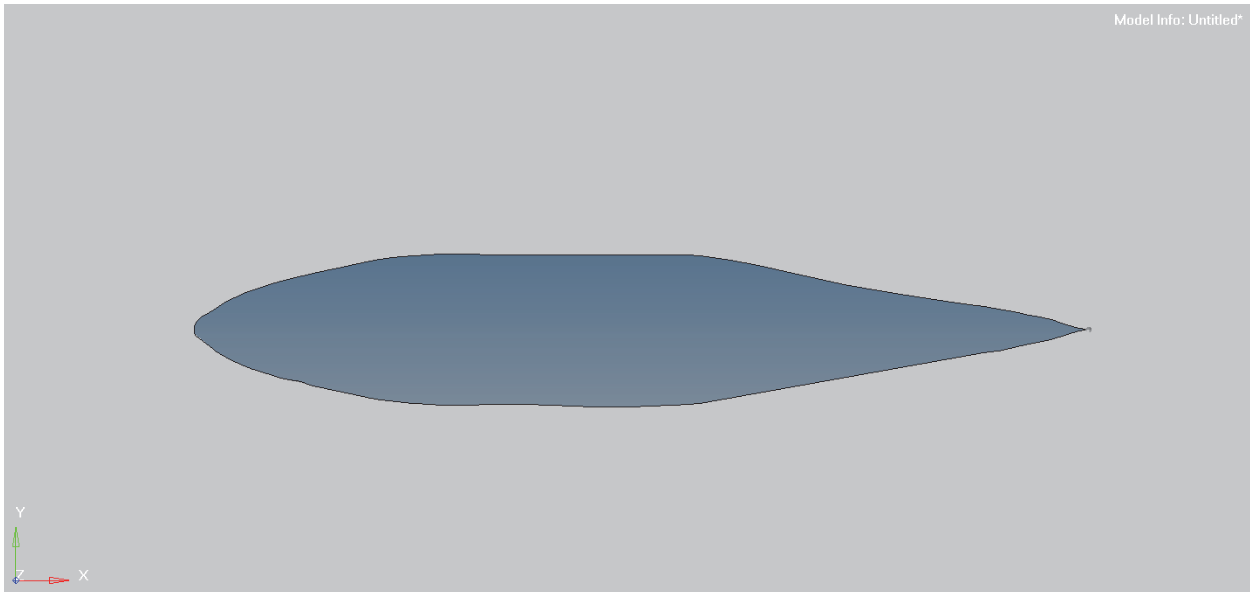

Figure 4. 2D Cross Section of the Tuna Model as Seen from Above

Turbulent fluid meshes generally consist of several layers of thin rectangular cells in the boundary layer regions and an unstructured grid in the far field. In Hypermesh, this is most effectively accomplished by growing elements out from the tuna model cross section to form the boundary layer cells. The far-field unstructured mesh was created and aligned with the boundary layer mesh to allow the two meshes to pass information to each other. The steps for doing this in Hypermesh are listed below. Again, it is assumed the user is already familiar with the program.

[Type the letter]o
mesh
Topology revision: keep mesh
graphics
geometry refinement: level 5

1D > line mesh
Select line outlining tuna cross section
Element Density = 0.1
reset element density to 400 elements

2D > drag Select the drag elems radio button Select all line elements z-axis
distance=0.1
on drag=1
offset +/− (you have to play with this to ensure the elements offset in the correct direction)

2D > elem offset
Select the solid layers radio button
Number of layers = 60
Total thickness = 0.02
Exponential Biasing Bias intensity = 0.75
CFD corners

tool>faces
elements-all
find faces

geom>lines
Manifold (closed linear)
Select "Trim input surface" radio button
Select main surface
Set to Node path
Select nodes around the outer boundary layer nodes that are coplanar with the XY plane

geom>point edit
Add on surfs
Nodes-node path

automesh trimmed and node respected surface
Element size = 0.1
type = Tri's

tool>edges
elems, select all
Tolerance = 0.00001
Preview Equivilance
Equivilance

Delete unwanted elements

The final mesh is shown in figure 5.

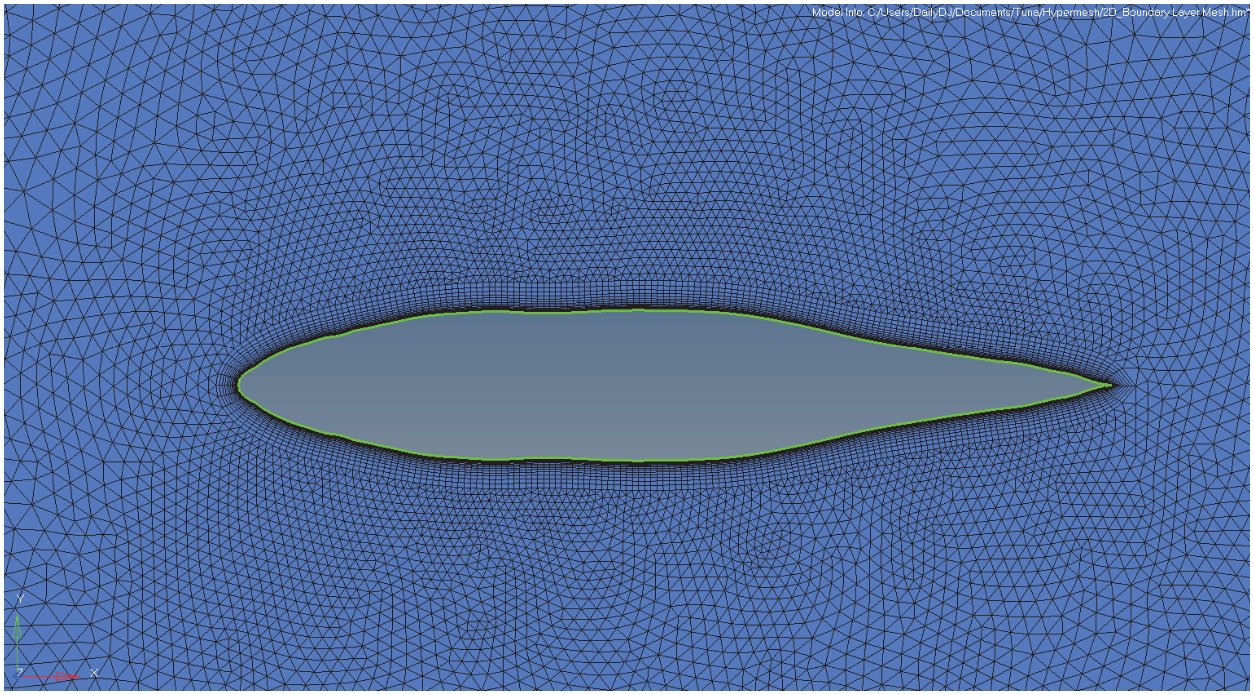

Figure 5. Final 2D Mesh for the Tuna Model including Boundary Layer Mesh

Bounding conditions were assigned to elements by selecting: **tool>edges>find edges**. This creates a new component named ^_*edges*, where the bounding elements for the inlet, outlet, walls, and tuna body were defined. Parameters for these elements were assigned later in the modeling process. The mesh was then exported as a Fluent 2D mesh.

2.1.4 3D Model

A 3D model was used to visualize the flow structures over the entire tuna body. This information was used to guide the selection of 2D measurement planes in the experimental and computational studies. However, a fully-turbulent, time-dependent solution in 3D required computational resources which were beyond the scope of this project. Therefore, to reduce the computational load of solving a fully turbulent model, a laminar flow field condition was prescribed. As such, the boundary layer elements could be less dense than in the 2D model intended for computations of the flow in a turbulent regime. The model was bisected vertically since the flow on the two halves of the model was considered to be symmetric. A semi-circle was drawn around the tuna model, rotated 20° about the x-axis, and then projected back onto the XY plane to create a semi-ellipse. A body of revolution was then created on half of the ellipse to create an ellipsoidal dome on one half of the model (shown in figure 6).

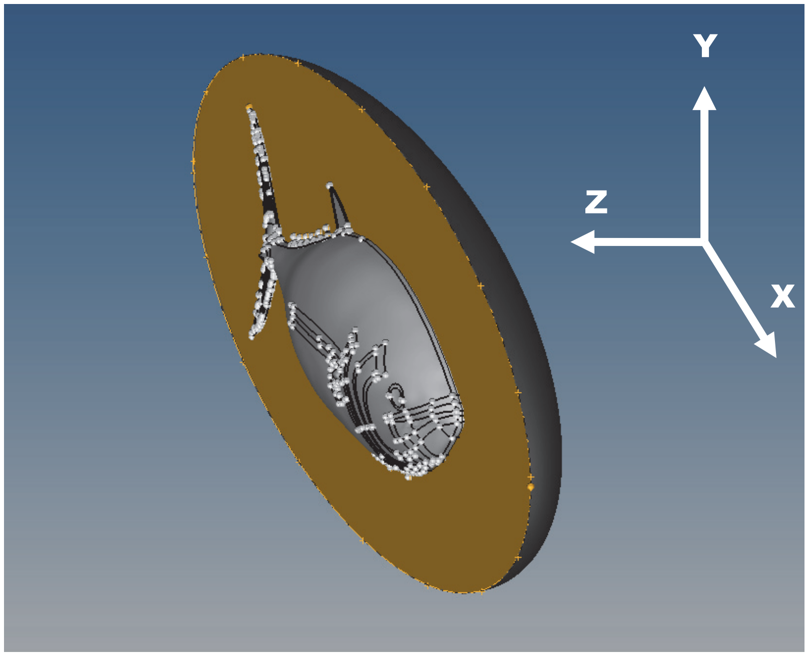

Figure 6. Semi-Ellipse Bisecting the 3D Tuna Model

A Boolean operation subtracted the tuna model from the ellipsoidal dome to create a tuna "mold." The surfaces of this mold were then meshed using the 2D automesh feature in Hypermesh (shown in figure 7).

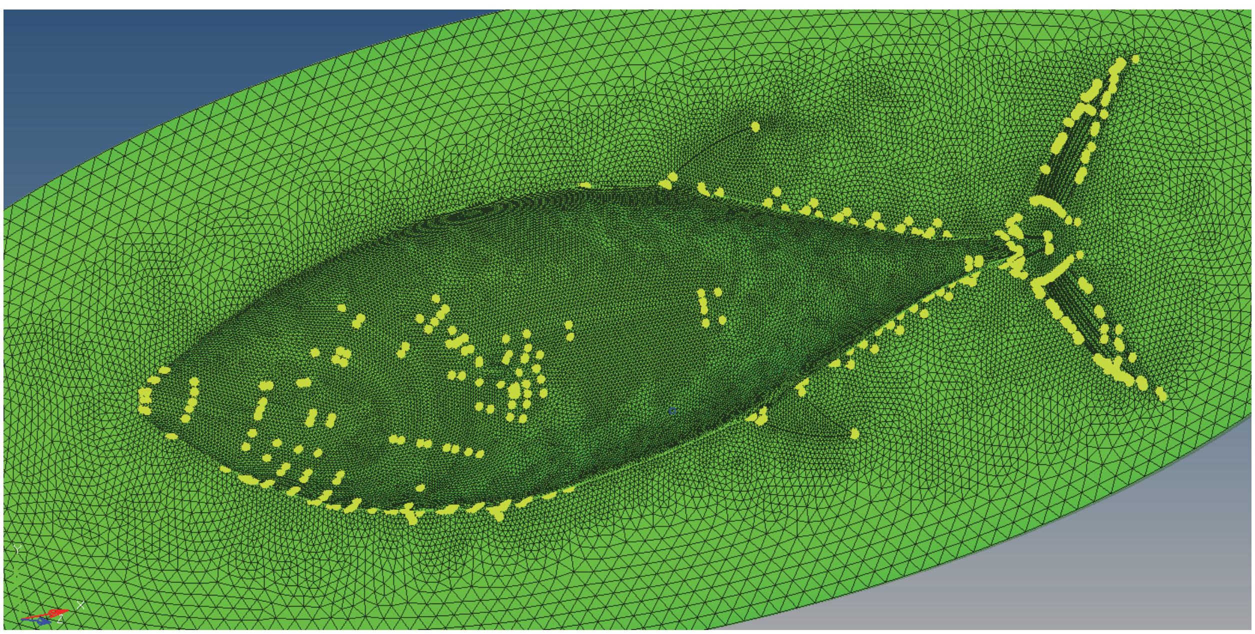

Figure 7. Surface Mesh of the Computationally-Generated 3D Tuna "Mold"

With the surfaces of the volume meshed, the entire volume between the surfaces was meshed using tetrahedral elements. Cross sections of the filled volume mesh are shown in figures 8a and 8b in horizontal and vertical planes. The elements increased in size from the surface of the tuna to the far field; this increased spatial resolution near the body while minimizing the computational demands of the model.

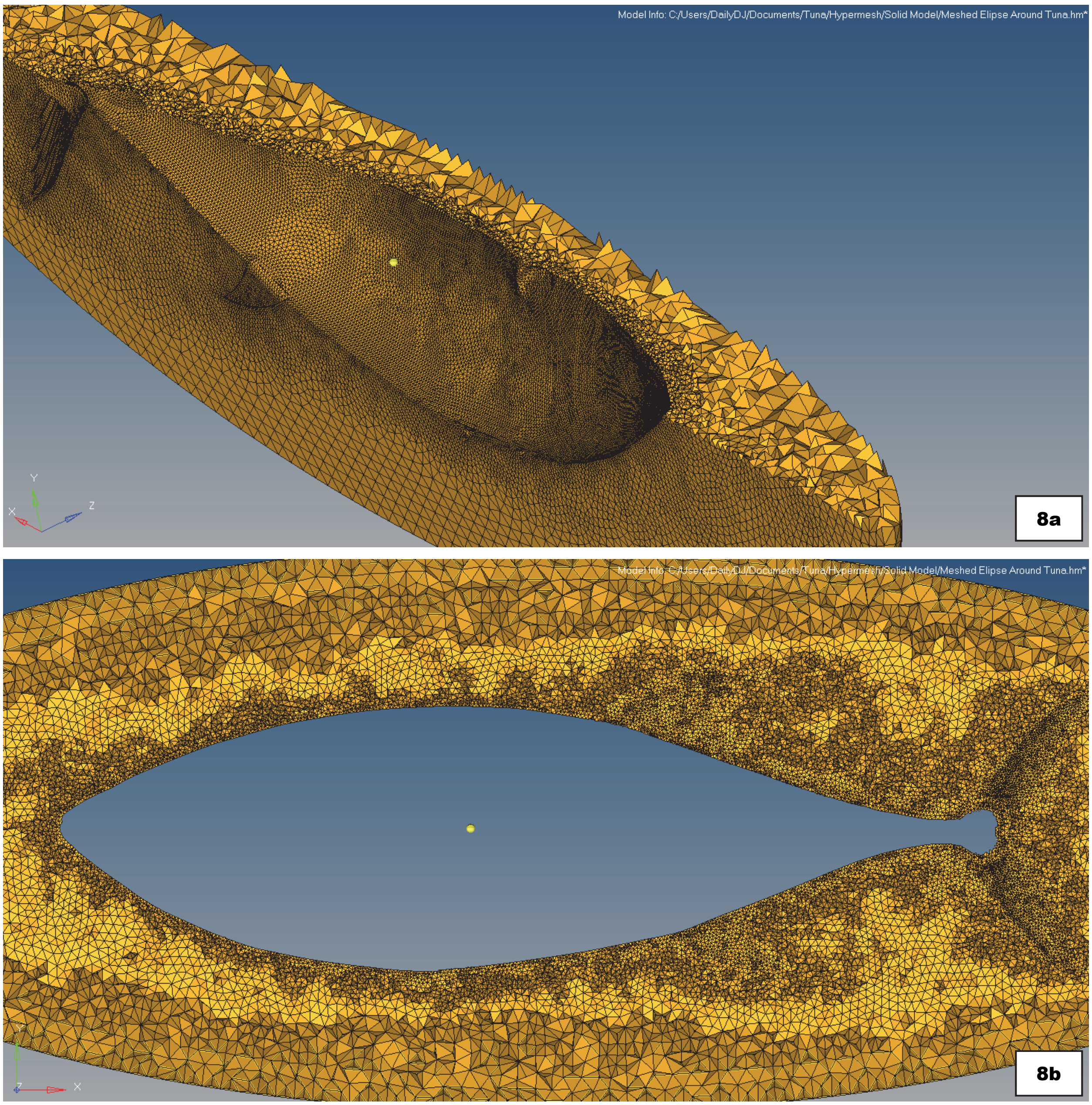

Figure 8. Cut-Away of the 3D Elements Surrounding the Tuna in (a) the Horizontal Plane (Isometric View) and (b) the Vertical Plane

A cube was then created around the ellipsoid and the ellipsoid was subtracted from the cube using a Boolean operation. The same meshing procedure was employed on the surface of the cube and the meshed interface between the cube and the ellipsoid were coupled to allow the two domains to communicate. Figure 9 is an isometric view showing a horizontal cross-sectional cut through the ellipsoid and the cube domains.

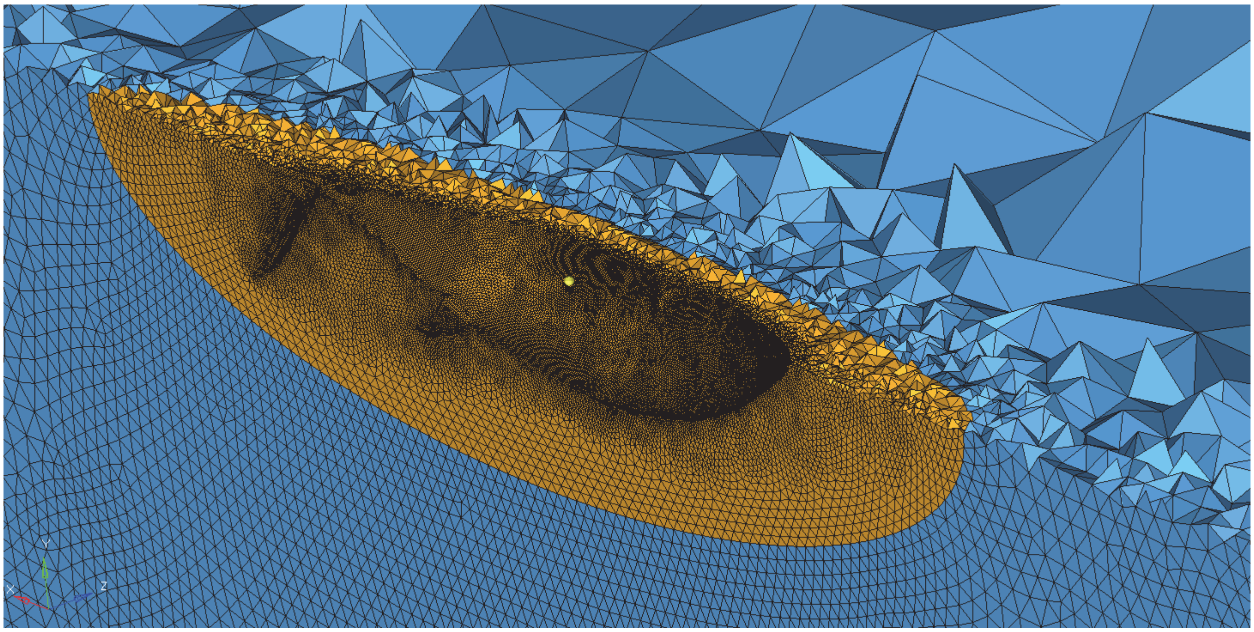

Figure 9. Near- and Far-Field 3D Meshes

Finally, boundary conditions were assigned to elements in a manner similar to the 2D model and the model was exported to Fluent as a 3D laminar model.

2.2 COMPUTATIONS

2.2.1 Input Parameters

The parameter settings for the 2D turbulent computational tuna model used in Fluent are outlined in table 1.

Table 1. Parameter Values for the 2D Model

1. GENERAL		
Type:	Pressure-Based	
Time:	Steady	
Velocity Formulation:	Absolute	
2D Space:	Planar	
2. MODELS		
Type:	k-epsilon	
k-epsilon Model:	Standard	
Near Wall Treatment:	Standard Wall Functions	
Model Constants:	Cmu:	0.09
	C1-Epsilon:	1.44
	C2-Epsilon:	1.92
	TKE Prandtl Number:	1
User Defined Functions:	Turbulent Viscosity:	None
	Prandtl Numbers:	None
3. MATERIALS		
Water-Liquid:	Density:	998 kg/m^3
	Cp:	0.9988
	Viscosity:	8.94E-4 Pa*s
4. CELL ZONE CONDITION		
Edit>>Material Name>>water liquid		
Operating Conditions>>x = −1, y = −1		
5. BOUNDARY CONDITIONS		
Inlet:	Velocity Magnitude:	10 m/s
	Turbulence Intensity (%):	5
	Turbulence Viscosity Ratio:	10
Tuna:	Stationary Wall:	No slip
	Wall Roughness:	0.5
Walls:	Stationary Wall:	Specified shear of 0

Table 1. Parameter Values for the 2D Model (Cont'd)

6. DYNAMIC MESH		
6.1 REFERENCE VALUES		
Compute From:	Fluid Domain	
Area:	0.2 m^2	
Density:	998 kg/m^3	
Length:	0.81 m	
Velocity:	10 m/s	
Viscosity:	8.94E-04 Pa*s	
Reference Zone:	Fluid Domain	
7. SOLUTION METHODS		
Pressure Velocity Coupling:	Simple	
Gradient:	Least Squares Cell-Based	
Pressure:	Standard	
Momentum:	Second Order Upwind	
Turbulent Kinetic Energy (TKE):	First Order Upwind	
Turbulent Dissipation Rate:	First Order Upwind	
8. SOLUTION CONTROLS		
Under-Relaxation Factors:	Pressure:	0.3
	Density:	1
	Body Forces:	1
	Momentum:	0.7
	TKE:	0.8
9. MONITORS		
Residuals:		
	Continuity:	1.00E-5
	X-Velocity:	1.00E-5
	Y-Velocity:	1.00E-5
	k:	1.00E-5
CD:	of tuna	
10. SOLUTION INITIALIZATION		
Hybrid Initialization:		
11. CALCULATION ACTIVITIES		
None		
12. RUN CALCULATION		
Number of Iterations:	1000	

A Reynolds-Averaged Navier-Stokes (RANS) k-epsilon solver was chosen for the 2D model due to the external flow nature of the problem. K-epsilon solvers are commonly used to solve flow fields for airfoils that bear a striking resemblance to the cross section of the Bluefin tuna The solver wall treatments and constants were left at the default settings. The working fluid was water and the inlet boundary condition was set to vary from 2 m/s to 22 m/s. These inlet velocities were chosen to mimic the approximate speeds tuna have reportedly been capable of achieving. The bounding walls of the model were slip walls while the surface of the tuna was a no-slip wall. Because the actual surface roughness of the tuna was neither known nor constant across its surface, the default surface roughness value of 0.5 was used on the model. All simulations were able to achieve sufficiently small residuals within 1000 iterations.

The 3D model used a laminar flow solver instead of a turbulent model due to the limited computational resources available. The model featured a velocity inlet boundary condition of 10 m/s (as a notional inlet condition) with additional parameter values listed in table 2.

Table 2. Parameter Values for the 3D Model

1. GENERAL		
Type:	Pressure-Based	
Time:	Steady	
Velocity Formulation:	Absolute	
3D:		
2. MODELS		
Viscous - Laminar		
3. MATERIALS		
Water-Liquid:	Density:	998 kg/m^3
	Cp:	0.9988
	Viscosity:	8.94E-4 kg/m-s
4. CELL ZONE CONDITION		
Edit>>Material Name>>water liquid		
Operating Conditions>>x = −1, y = −1		
5. BOUNDARY CONDITIONS		
Inlet:	Velocity Magnitude:	10 m/s
Tuna:	Stationary Wall:	No slip
	Wall Roughness:	0.5
Walls:	Stationary Wall:	Specified shear of 0

Table 2. Parameter Values for the 3D Model (Cont'd)

6. DYNAMIC MESH		
6.1 REFERENCE VALUES		
Compute From:	Fluid Domain	
Area:	0.2 m^2	
Density:	998 kg/m^3	
Length:	0.81 m	
Velocity:	10 m/s	
Viscosity:	8.94E-04 kg/m-s	
Reference Zone:	Fluid Domain	
7. SOLUTION METHODS		
Pressure Velocity Coupling:	Simple	
Gradient:	Least Squares Cell-Based	
Pressure:	Standard	
Momentum:	Second Order Upwind	
Turbulent Kinetic Energy:	First Order Upwind	
Turbulent Dissipation Rate:	First Order Upwind	
8. SOLUTION CONTROLS		
Under-Relaxation Factors:	Pressure:	0.3
	Density:	1
	Body Forces:	1
	Momentum:	0.7
	TKE:	0.8
9. MONITORS		
Residuals:		
	Continuity:	1.00E-5
	X-Velocity:	1.00E-5
	Y-Velocity:	1.00E-5
	k:	1.00E-5
CD:	of tuna	
10. SOLUTION INITIALIZATION		
Hybrid Initialization:		
11. CALCULATION ACTIVITIES		
11.1 RUN CALCULATION		
Number of Iterations:	1000	

2.2.2 Grid Convergence

Grid convergence for the 2D turbulent model was verified using the built-in Fluent grid refinement algorithms. The original mesh created in Hypermesh is shown in figure 10. This mesh was solved using the parameter values specified in the previous sections. The mesh was adapted by calculating the gradients of pressure, velocity, nondimensional wall units (y+), and turbulence intensity. A common strategy for grid refinement is to mark and refine elements that are within 10% of the maximum calculated gradient.

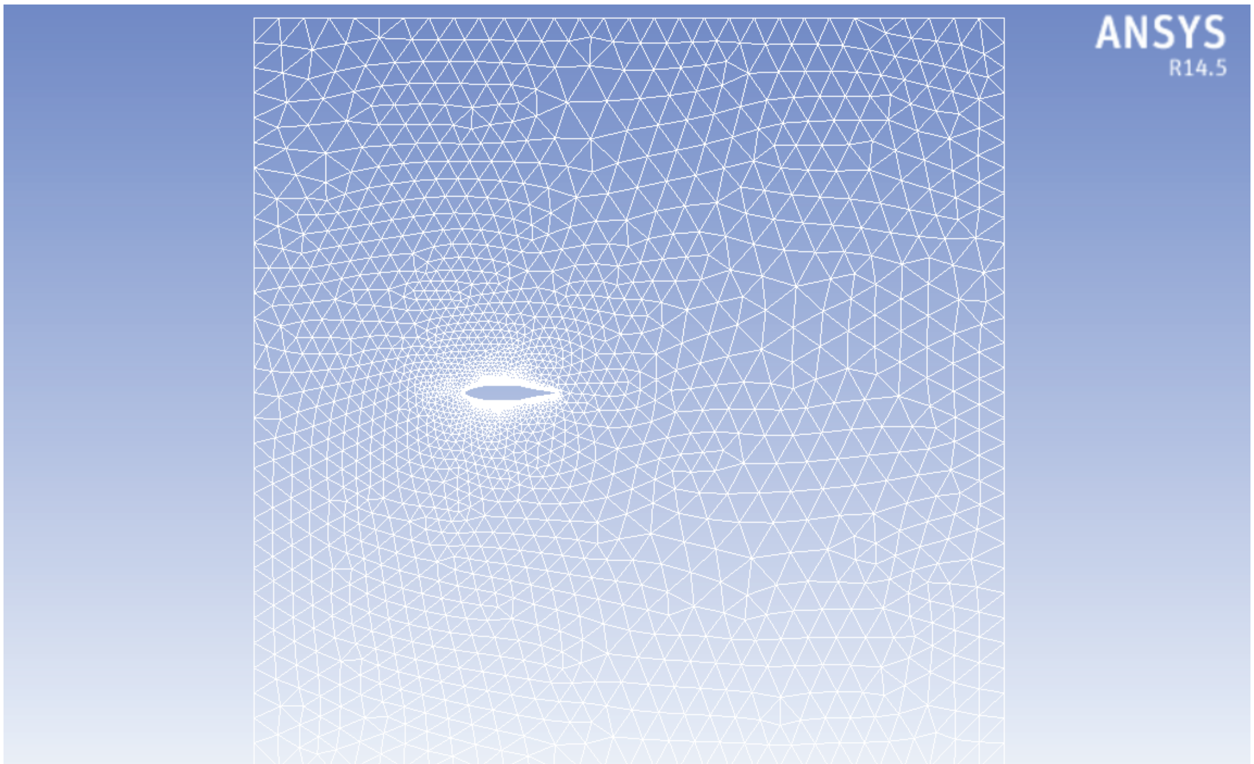

Figure 10. Original Mesh Created in Hypermesh

Figure 11 shows the result when the pressure gradient was used to mark elements for refinement. The maximum calculated static pressure gradient was found to be 4610 Pa/m. The refined threshold was set to approximately 10% of that value, or 400. The cells that fall within that threshold are flagged in red. Based on this outcome, Fluent generated a new mesh with elements added in the higher-gradient regions. This mesh was used as a starting point to continue iterating on the size, shape, and number of elements to determine grid independence.

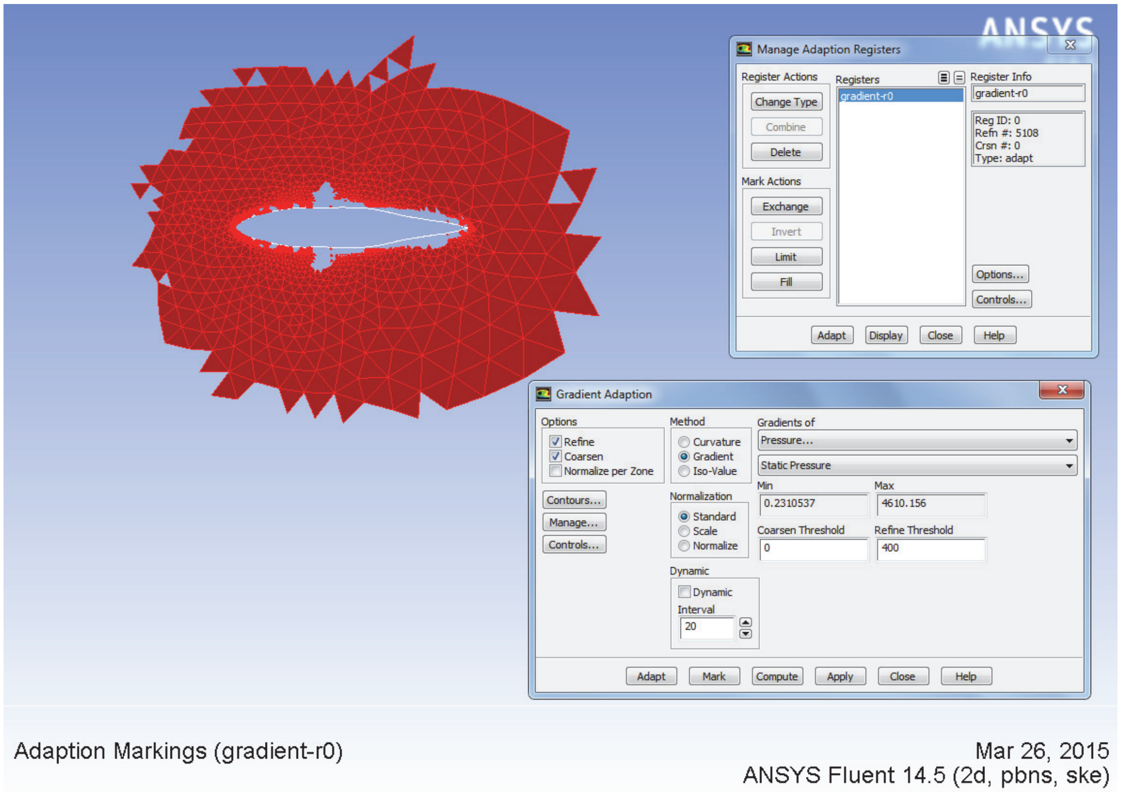

Figure 11. Screenshot Showing Elements Marked for Grid Refinement

The coefficient of drag was the primary metric used to monitor convergence from one model to the next along with residuals of continuity, velocity, and turbulence as a means to monitor the stability of each solution. These values were tracked through the iterations of mesh refinement and are recorded in table 3.

Table 3. Parameters Used to Monitor Grid Convergence

		Mesh 1	Delta	Actions taken	Mesh 2	Delta	Actions taken	Mesh 3	Delta	Actions taken
Number of iterations		1000	None	Mesh was adapted using pressure gradient threshold of 400. The maximum gradient was 4610. Mesh was adapted using Wall yplus gradient threshold of 10. The maximum gradient was 97.	1000		Mesh was adapted using pressure gradient threshold of 200. The maximum gradient was 2685. Mesh was adapted using turbulence intensity gradient threshold of 0.01. The maximum gradient was 0.13. The mesh downstream of the tuna was further refined by selecting a region manually for refinement.	1000		Mesh was manually adapted downstream of the tuna model.
Residuals	Continuity	5.8268E-05			1.3959E-04	8.1322E-05		5.5219E-05	2.6103E-05	
	x-velocity	1.1801E-07			1.7999E-07	6.1980E-08		1.0520E-06	9.9002E-07	
	y-velocity	8.2424E-08			1.2778E-07	4.5356E-08		2.8800E-07	2.42644E-07	
	k	3.4070E-07			1.2080E-05	1.1739E-05		3.1282E-05	1.95427E-05	
	Epsilon	4.2979E-07			5.8362E-05	5.7932E-05		1.2926E-05	4.50062E-05	
	Cd	4.5610E-02			4.3585E-02	2.0250E-03		4.6893E-02	0.044868	
Number of cells		16360			33259	1.6899E+04		1.1630E+05	99403	
Number of Faces		30750			63036	3.2286E+04		2.2183E+05	189546	
Number of nodes		14390			29777	1.5387E+04		105530	90143	

		Mesh 4	Delta	Actions taken	Mesh 5	Delta	Actions taken	Mesh 6	Delta	Actions taken
Number of iterations		1000		Mesh was adapted using Wall Yplus gradient threshold of 5. The maximum gradient was 47. The mesh downstream of the model was manually refined to smooth the velocity and turbulence intensity gradients.	1000		Mesh was adapted downstream of the flow field using manual mesh adaptation.	1000		None
Residuals	Continuity	6.7090E-05	4.0987E-05		6.5761E-05	2.4774E-05		6.6260E-05	0.000041486	
	x-velocity	1.3936E-06	4.0358E-07		2.8262E-06	2.4226E-06		2.8564E-06	4.3378E-07	
	y-velocity	3.6575E-07	1.2311E-07		4.9574E-07	3.7263E-07		5.0787E-07	1.35236E-07	
	k	4.1409E-05	2.1866E-05		6.7848E-05	4.5982E-05		6.8493E-05	2.25113E-05	
	Epsilon	1.4153E-05	3.0853E-05		2.8760E-05	2.0932E-06		2.2765E-05	2.06718E-05	
	Cd	4.6965E-02	2.0970E-03		5.6767E-02	5.4670E-02		5.6733E-02	0.002063	
Number of cells		120328	20925		144256	123331		145624	22293	
Number of Faces		228652	39106		271445	232339		273518	41179	
Number of nodes		108324	18181		127189	109008		127894	18886	

The first four meshes show a C_d value between 0.43 and 0.469, but inspection of the velocity profiles indicated a region downstream of the model that required further refinement. The grid was refined manually in the specified region and the C_d value changed to 0.546. This value did not change significantly with subsequent mesh refinement and the model was therefore considered to be converged (see figure 12).

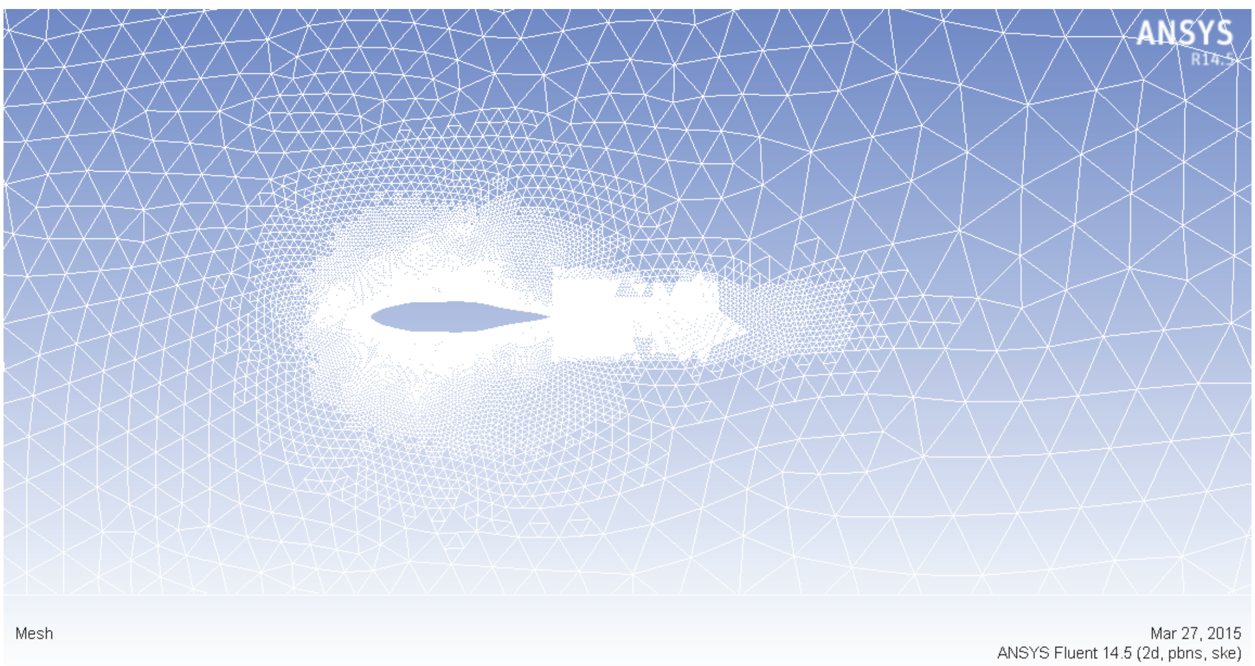

Figure 12. Final Mesh as Refined by Fluent

The 3D model did not undergo mesh refinement since the purpose of the 3D model was primarily to show gross flow structures rather than to compute details of the boundary layer. Therefore, the 3D model was given an initial mesh that was considered to be sufficiently fine and no adaptation scheme was applied.

3. RESULTS

3.1 2D STEADY-STATE

The model was run for 1000 iterations and a plot of velocity contours with a 10 m/s inlet velocity condition is shown in figure 13. The contours of velocity are symmetric about the longitudinal axis of the tuna model. The areas of maximum velocity occur in two lobes located at the midbody of the tuna. Lobes of lower velocity appear just forward of the caudal fin and persist until after the caudal fin. The exact reason for this decreased area of velocity is not currently known and requires further investigation. The detailed experimental velocity measurements will help characterize the flow in that region.

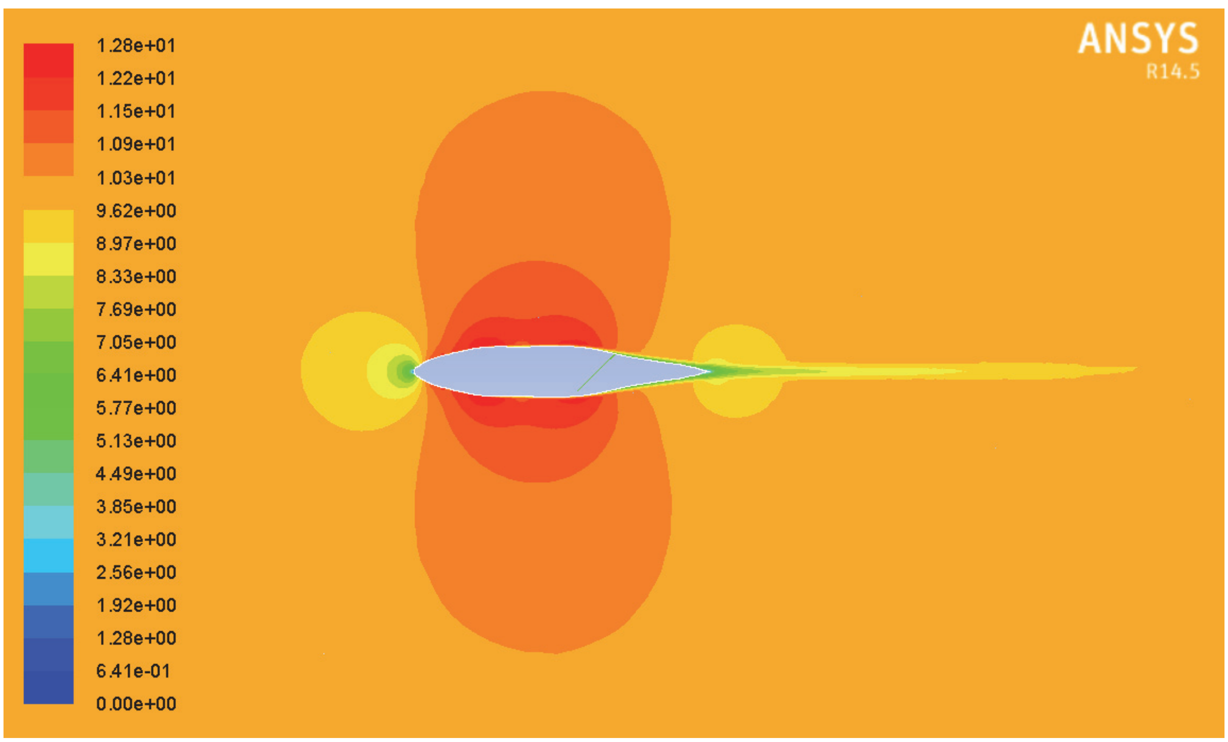

Figure 13. Contours of Velocity Shown in m/s

A plot of the turbulence intensity contours in units of m/s is overlaid with the mesh in figure 14. In this plot it is apparent that the additional mesh elements downstream of the model were required to capture the turbulence intensity created in the wake of the tuna model.

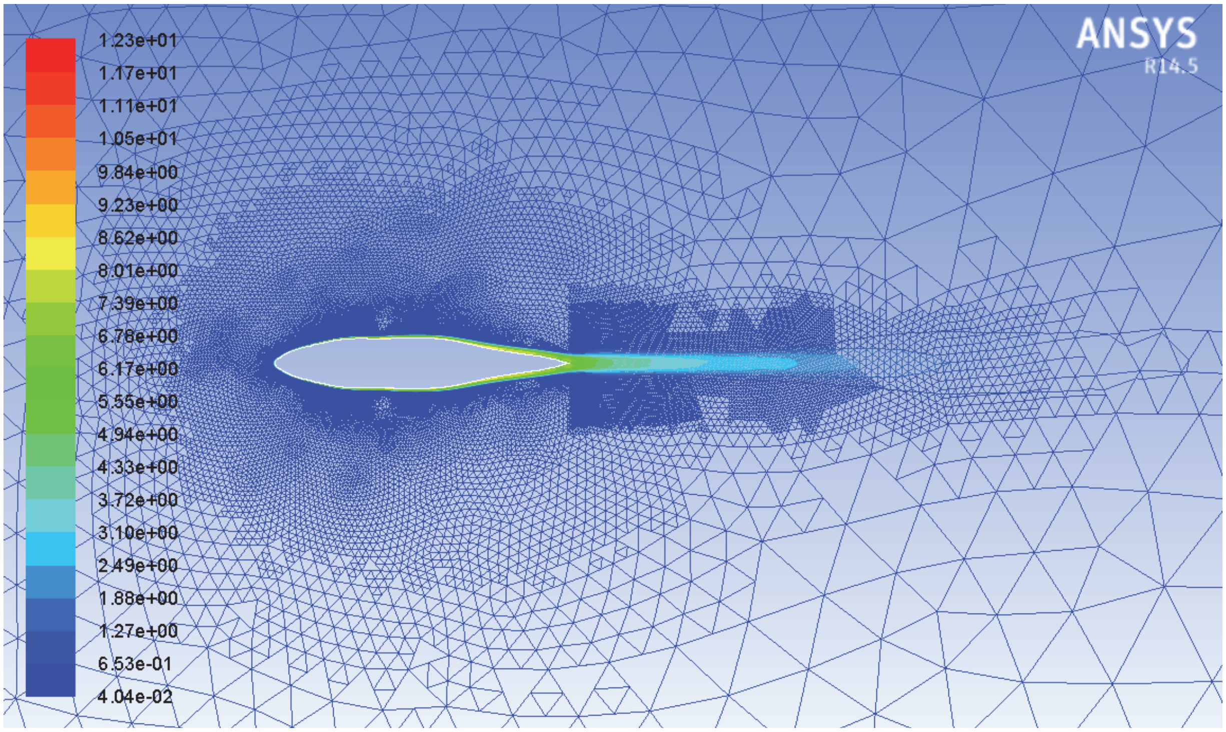

Figure 14. Contours of Turbulence Intensity in m/s Overlaid with the Mesh

A plot of the nondimensional parameter y+ in the favorable pressure gradient region of the model is shown in figure 15. Values of y+ on the order of unity can be used as an indication that there are an adequate number of elements near the wall to resolve the boundary layer. The results in figure 15 show that y+ values of approximately three were achieved, which was considered an acceptable value.

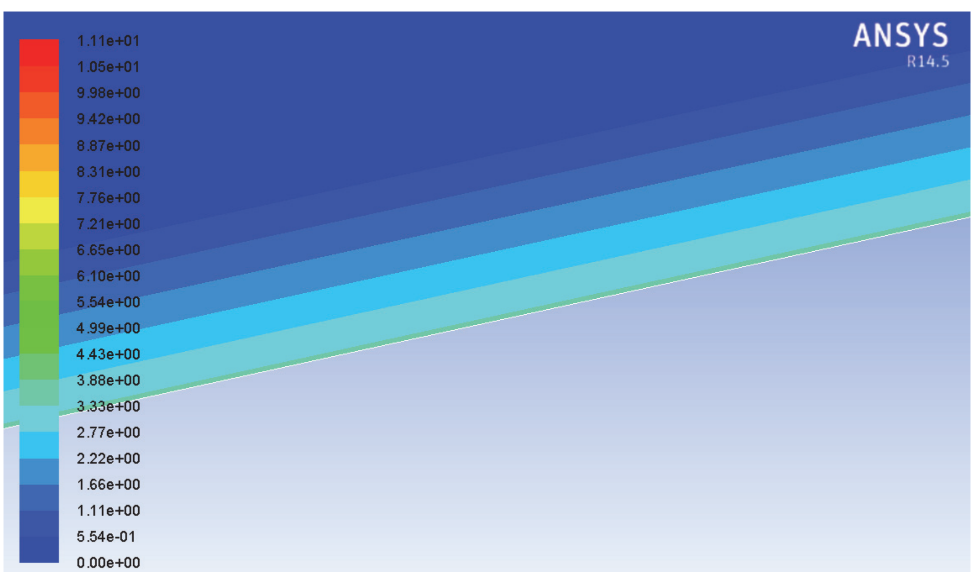

Figure 15. Contours of Wall y+

3.1.1 Boundary Layer Characteristics

The boundary layer characteristics along the surface of the tuna model were determined by observing the velocity profiles of fifteen surface normals along the surface of the tuna model, as shown in figure 16.

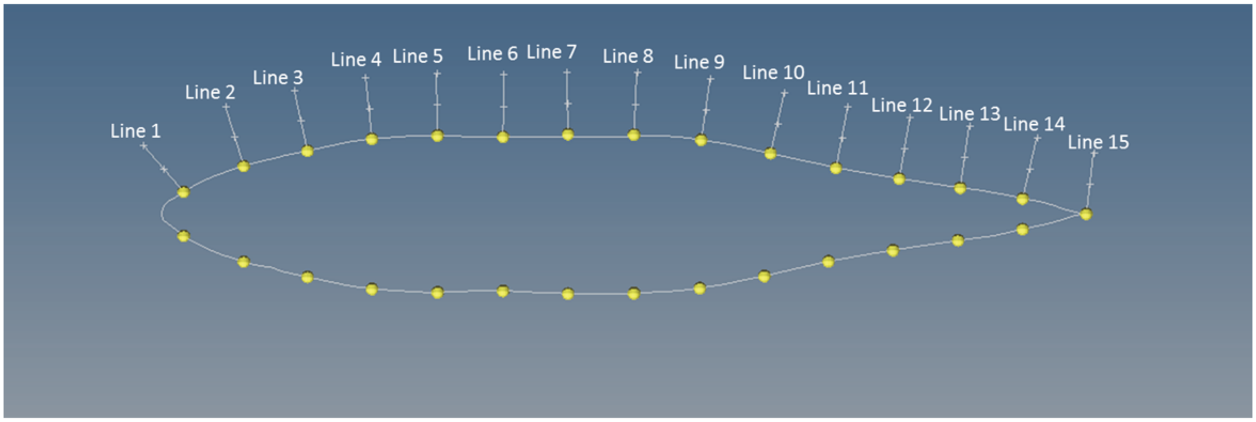

Figure 16. Lines Normal to the Surface of the Tuna Model for Plotting the Boundary Layer Velocity

Velocity values along the surface normals were computed and plotted in figure 17 along each of the surface normal lines for inlet velocity conditions from 2 to 22 m/s (in 2 m/s increments).

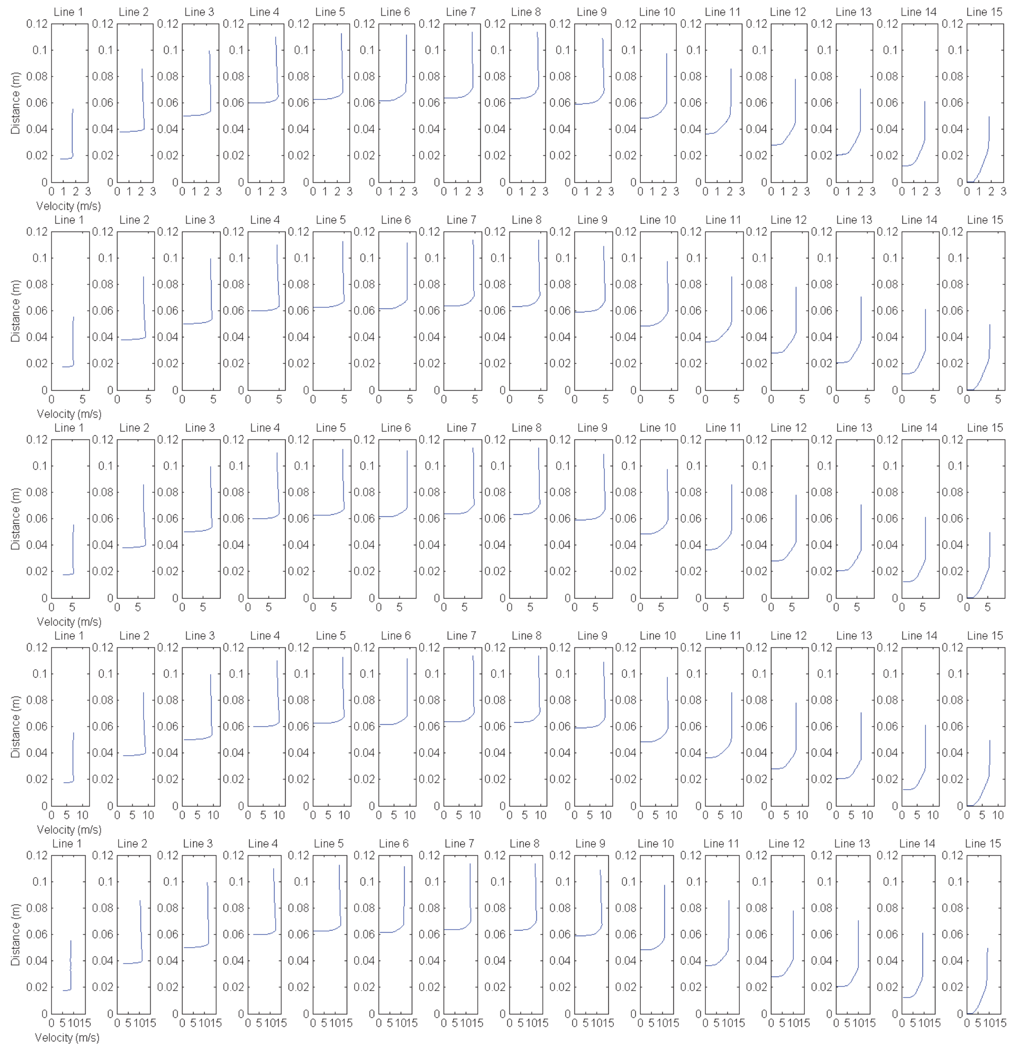

Figure 17. Velocity Profiles in m/s along the Lines Normal to the Surface of the Tuna Body from 2 to 22 m/s in 2 m/s Increments

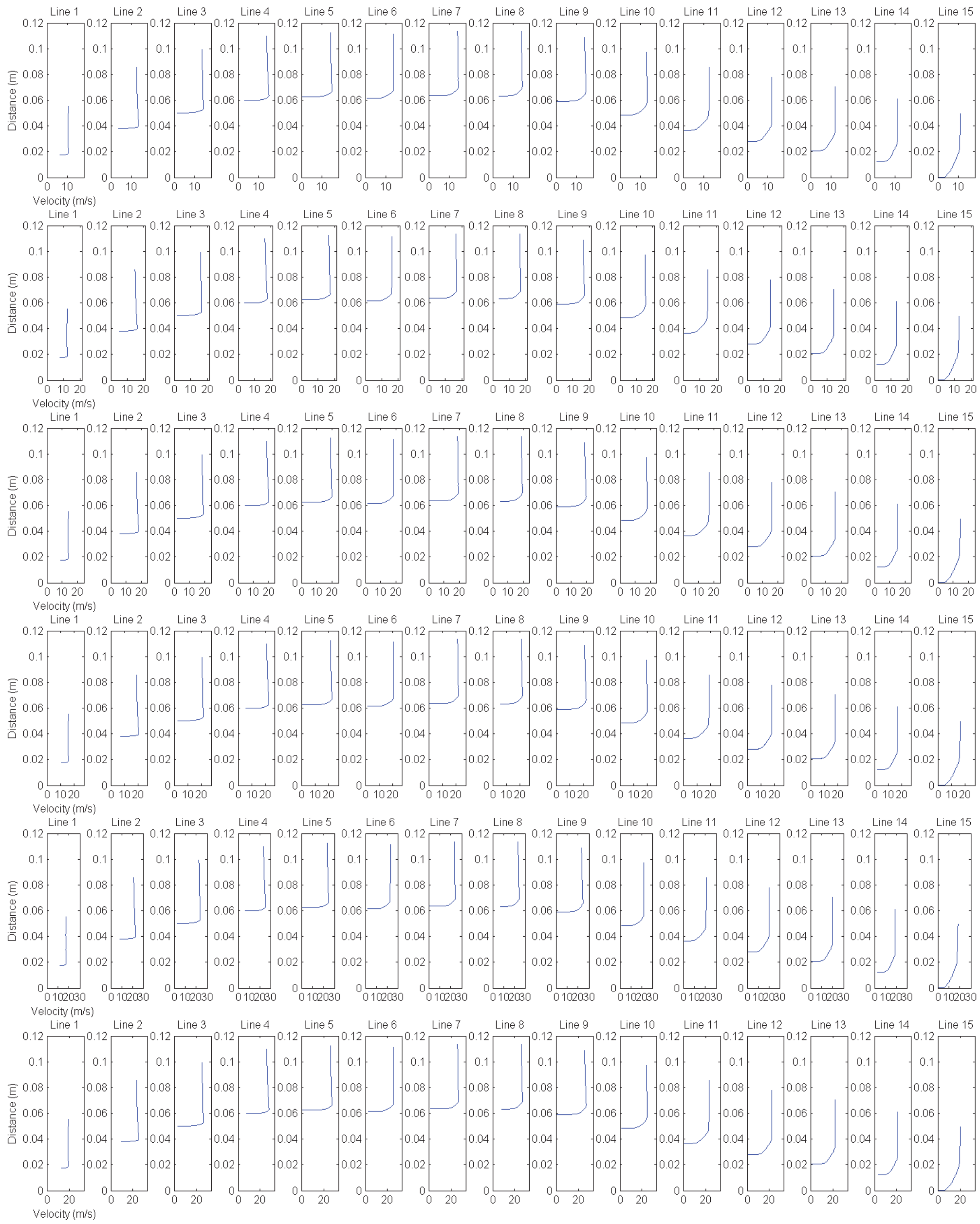

Figure 17. Velocity Profiles in m/s along the Lines Normal to the Surface of the Tuna Body from 2 to 22 m/s in 2 m/s Increments (Cont'd)

The velocity profiles were then overlaid for each velocity case. These profiles show the shape of the boundary layer more clearly (see figure 18).

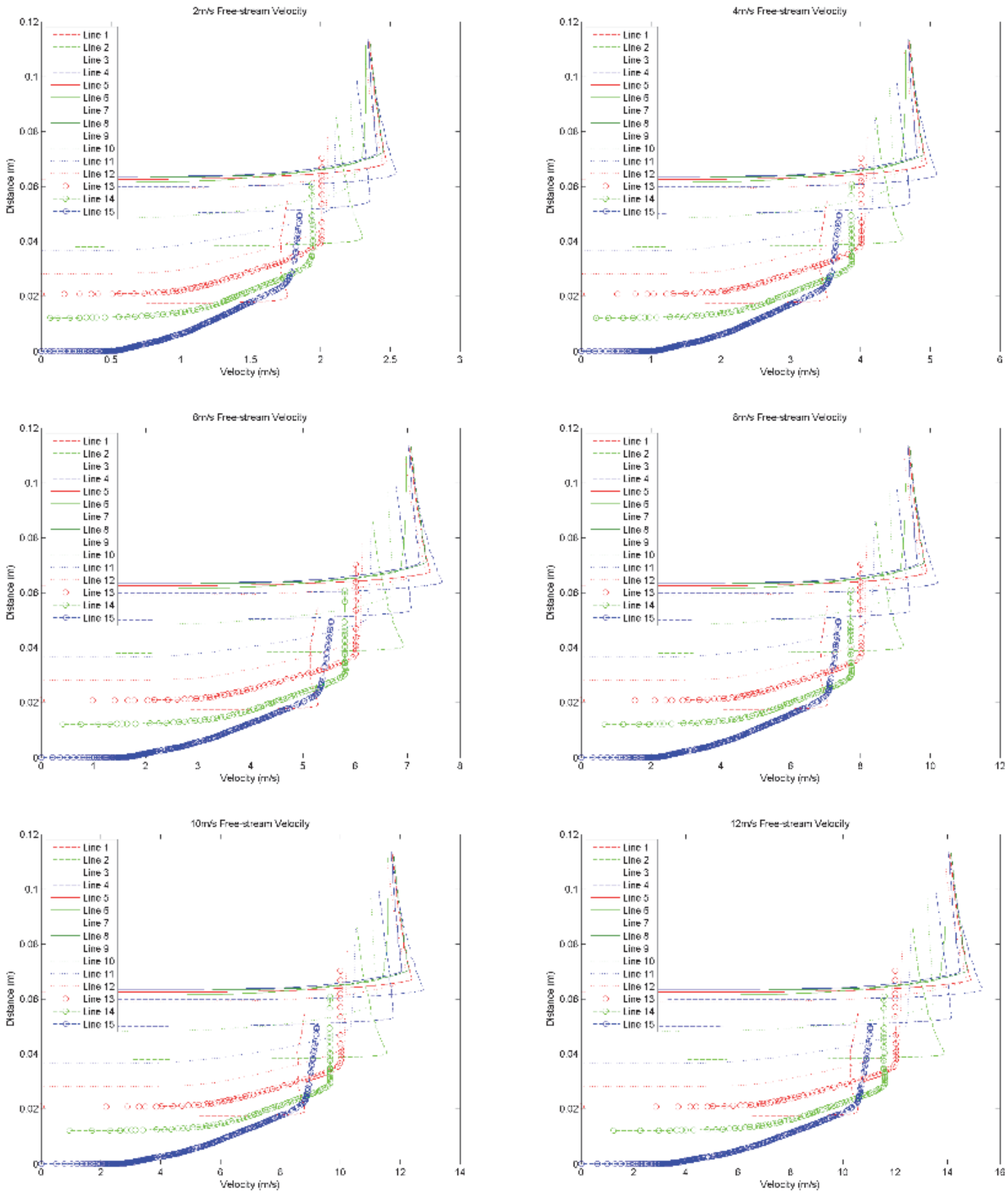

Figure 18. Velocity Profiles in m/s along the Lines Normal to the Surface of the Tuna Model at each Inlet (speed from 2 to 22 m/s)

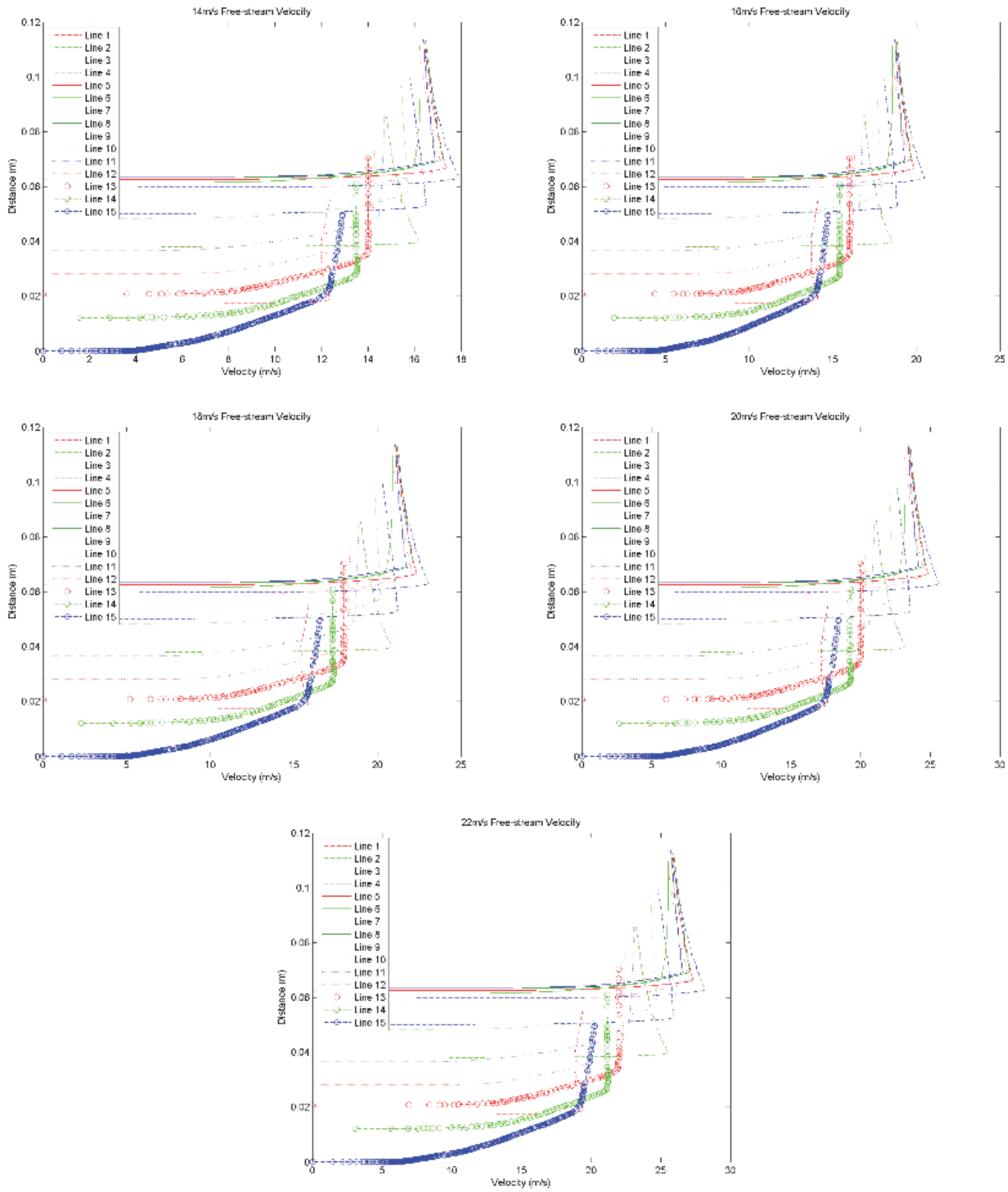

Figure 18. Velocity Profiles in m/s along the Lines Normal to the Surface of the Tuna Model at each Inlet (speed from 2 to 22 m/s) (Cont'd)

3.2 2D TRANSIENT

The 2D case was also run as a transient model to compute the fluctuating velocity components in the boundary layer. The first calculations were performed at 10 m/s with smooth walls and using the four-equation transition version of the k-epsilon turbulence model. The fluctuating velocity profile was plotted at several locations along each of the 15 normal lines

along the tuna model to determine if the boundary layer was transitioning to turbulent flow (see figure 19). Unfortunately, this method for visualizing the fluctuating component of the boundary layer is ambiguous. However, the results seem to indicate that the boundary layer remains laminar at 10 m/s with a smooth wall function. A more definitive method of identifying laminar or turbulent flow is being pursued. In addition, the default values for surface roughness are not expected to accurately represent the surface of the tuna and this parameter could impact the transition location. Related experimental measurements are ongoing and will be used to determine a more appropriate value of the surface roughness.

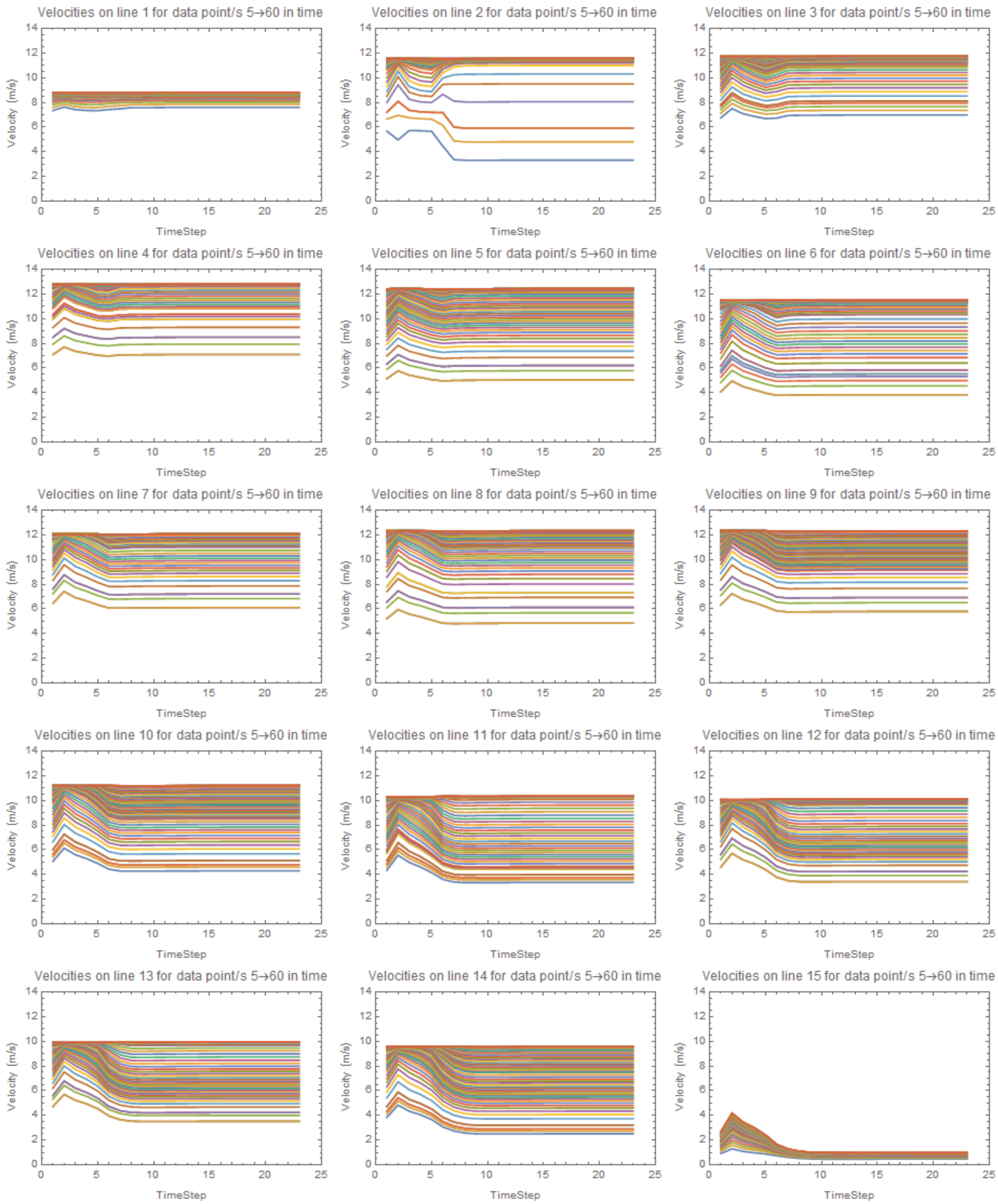

Figure 19. Fluctuating Velocity Profiles Computed at 15 Normal Lines for an Inlet Velocity Value of 10 m/s

3.3 3D STEADY-STATE MODEL

As previously stated, the 3D model was intended primarily for qualitative flow visualization rather than for quantitative research; therefore, the 3D computations were performed as laminar rather than turbulent flow. The 2D model of the tuna represented a cut at the centerline of the 3D tuna model. If the 3D flow around the tuna showed streamlines passing out-of-plane at the centerline, this would indicate that the centerline plane was a poor choice for modeling 2D flow. Analyzing the results from the 3D computations to ensure that streamlines were not passing through the computational plane validates the use of a 2D model. 2D computations should be performed where the flow was mostly planar.

Two methods of flow visualization were employed to show the flow field around the tuna model for an inlet velocity of 10 m/s: oil flow visualization lines (figure 20), and particle tracers emitted from the surface of the model (figure 21). In both plots, the color scales on the left-hand side indicate the velocity values in m/s. In figure 20, the model is positioned in a slightly isometric view but the computations were performed at zero pitch and yaw angles. No significant three-dimensional flow was apparent from either method.

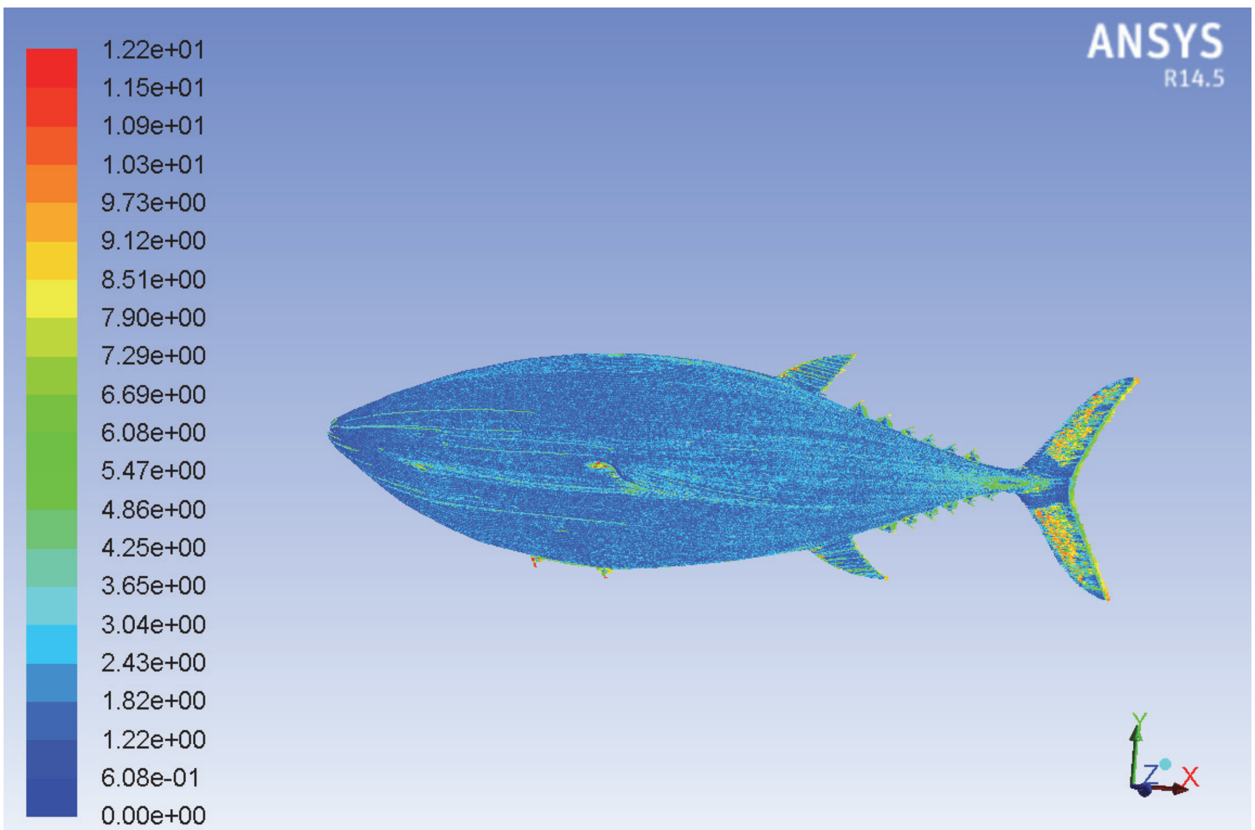

Figure 20. Oil Flow Lines from the 3D Computations on the Tuna Model

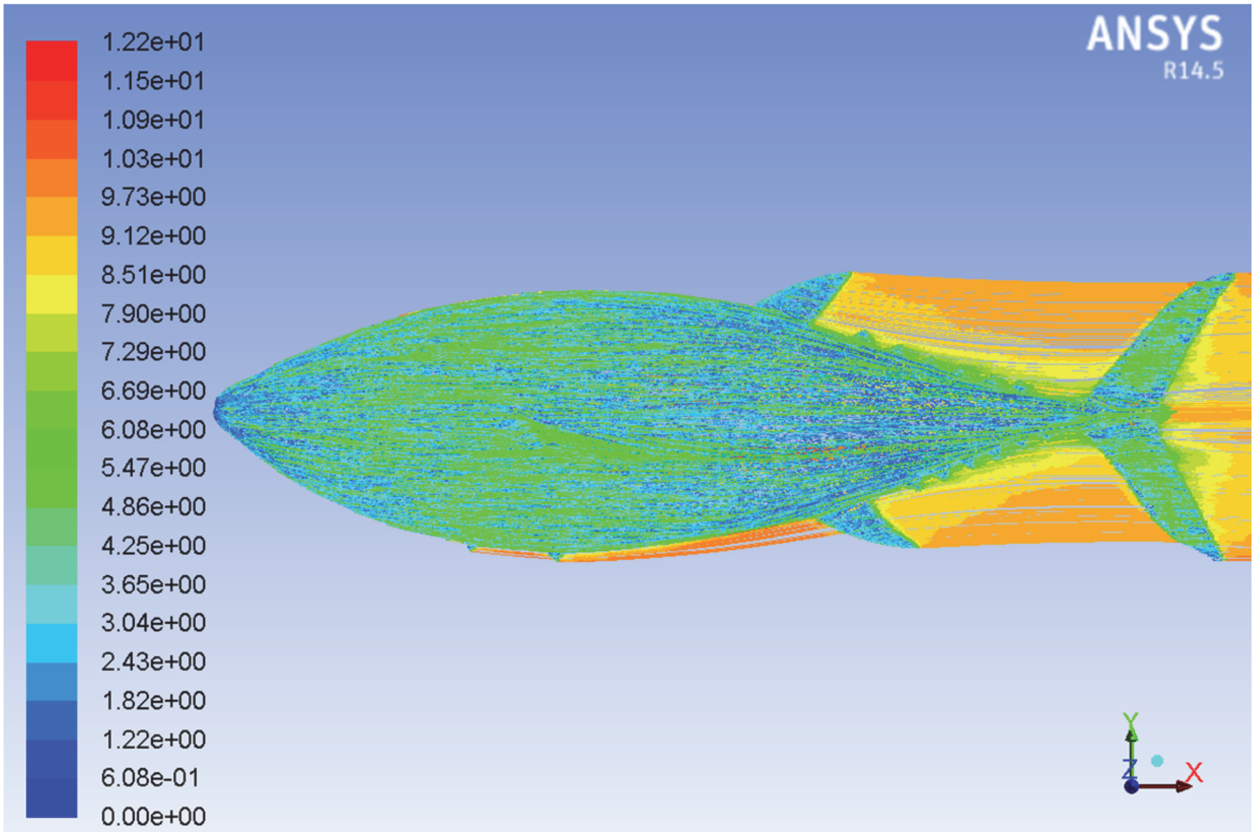

Figure 21. Particle Tracers Emitted from the Surface of the Tuna Model

The static pressure on the surface of the model was also plotted and is shown in figure 22. It is interesting to note that the static pressure increases in the adverse pressure gradient region of the model. This increase in pressure coincides with the location of the small finlets along the superior and inferior spines of the tuna and a distinct change in the surface roughness of their scales in this region (reference 2).

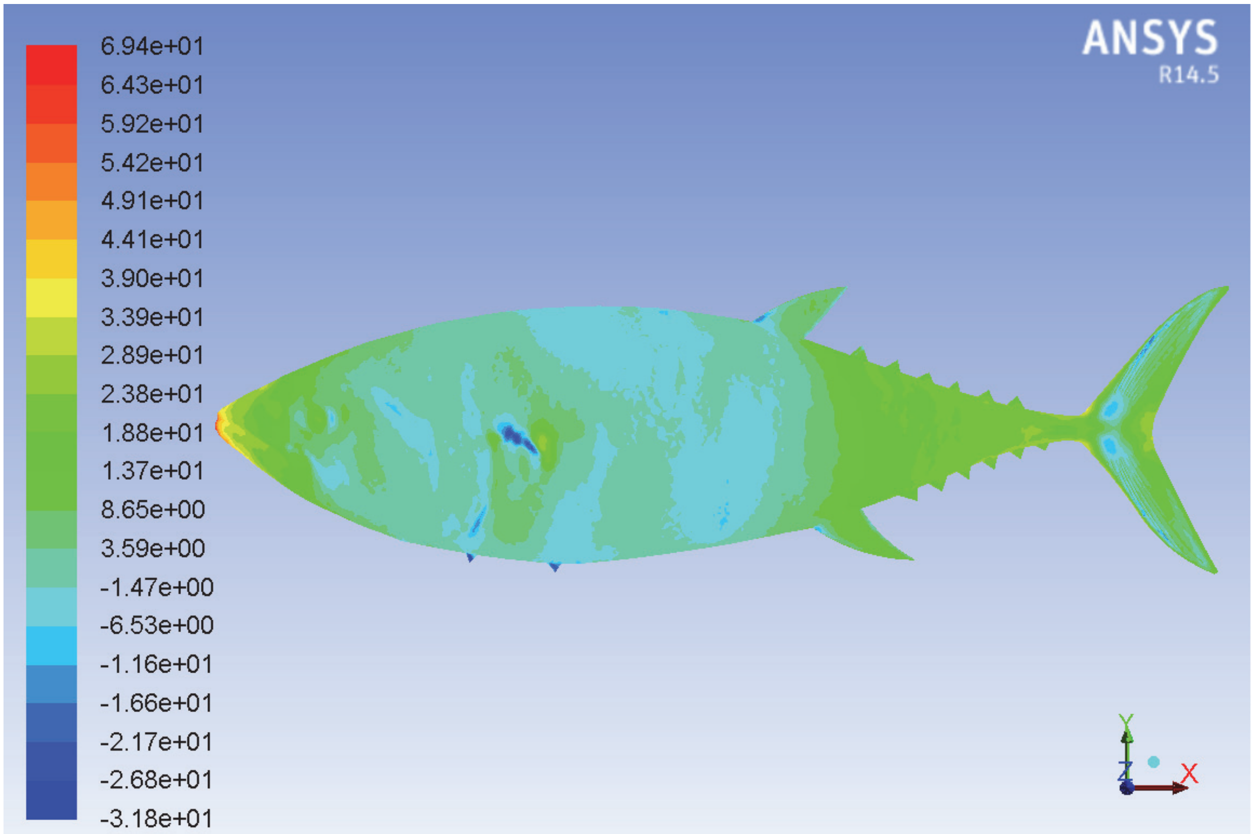

Figure 22. Contours of Pressure on the 3D Tuna Model

4. DISCUSSION AND ONGOING WORK

The shape of the velocity profiles from the 2D model suggests that the boundary layer transitions to turbulent flow by the time it reaches Line 2, or 11% chord. This result did not vary significantly with speed. By contrast, the experimental results obtained during experiments in the NUWC Division Newport tow tank suggest that the boundary layer remains laminar up to 3 m/s (maximum operating speed of the facility). The recently-conducted experiment in the NSWC Carderock 36-inch water tunnel will further contribute to this analysis. The fluctuating velocity components were also calculated from the 2D model, but did not provide a clear indication of whether the boundary layer was laminar or turbulent. The lack of agreement between the computational and experimental results is likely due to the inaccuracy of the value of the surface roughness parameter used in the computational model. A more appropriate value will be gleaned from comparison with detailed experimental boundary layer measurements.

The streamlines from the 3D model show a vertical flow component near the pectoral fin. However, this result is assumed to be an artifact caused by the pectoral fins being removed prior to casting the mold and is not necessarily representative of the flow in this region on an actual tuna body. Nevertheless it is assumed that it would have a minimal effect on the development of the boundary layer upstream of that region.

Velocity measurements on a full-scale experimental model of a Bluefin tuna have recently been completed and will be used to validate the computational results. Comparison of the results from the computational and experimental investigations will allow refinement of the input parameter values for future CFD calculations. Once the computational model has been fully validated, it will serve as an invaluable tool to significantly extend the speed range beyond what is feasible experimentally as well as facilitating investigations into the effects of the fins.

Finally, this research has the potential to advance technology of various Navy systems, e.g., torpedo and autonomous underwater vehicle (AUV) drag reduction by control surface manipulation, novel shapes, efficient glider designs, flow noise mitigation for towed arrays, and advancements in bio-inspired propulsion. The plan is to continue these investigations in collaboration with Stanford University, Massachusetts Institute of Technology, and the University of Rhode Island.

REFERENCES

1. Kimberly M. Cipolla, *Flow Measurements on Bluefin Tuna Shape*, Ocean Battlespace Sensing, ONR 321MS FY-13 Annual Reports, October 2013.

2. Kimberly M. Cipolla, "Characterization of the Boundary Layers on Full-Scale Bluefin Tuna," NUWC-NPT Technical Report 12,163, Naval Undersea Warfare Center, Newport RI, 30 September 2014.

DISTRIBUTION LIST

External
Office of Naval Research 321MS (C.M. Traweek)
Defense Technical Information Center (Input Support Branch)
Center for Naval Analyses
Georgia Institute of Technology (D. Trivett)
University of Rhode Island (J. Dahl)
Stanford University (B. Block)

Internal

Codes:	1033	(Science and Technology Library)
	1522	(B. Amaral (6))
	1512	(D. Daily (9))
	00T1	(K. Cipolla (6))
	00T1	(N. Dubois)
	00X	(D. Potvin)
	07SP	(J. Nasser)
	60X	(S. Hester)
	85	(C. Henoch)

Total: 33